International Journal of Advanced Research and Development
ISSN: 2455-4030
www.newresearchjournal.com/advanced
Volume 1; Issue 2; February 2016; Page No. 01-05

Design and analysis of plastic Mould for a hand guard

[1] Vikram yadav, [1] A Sunith kumar, *[1] M Devander, [2] S Phaneendra
[1] Department of Mechanical Engineering, SVS Group of Institutions, Warangal, Telangana, India.
[2] Department of Mechanical Engineering, BITS, Warangal, Telangana, India.

Abstract
Now a day's plastics are used in many of the component and they are replacing many metals in some of the engineering applications. Major advantages of using plastics are formability and they provide a low cost to the performance ratio. The different processes used to fabricate plastic components are extrusion, injection moulding, blow moulding, rotational moulding, and compression moulding, by far the most used process for plastics is injection moulding. Plastic injection moulding is one of the most popular manufacturing processes for making thermoplastic products, and mold design is a key aspect of the process. Design of moulds requires knowledge, expertise and most importantly experience in the field. For this work, the project is to design a plastic mould for Hand Guard. The design is made considering the manufacturing resources and ability of the company within minimum possible time and at least cost of manufacturing. The tool should be able to accommodate minor modifications. The component is designed and modeled in 3-D by using PRO-E software and then analysis of the component is performed in the Mold Flow Plastics Advisers. And detailed design procedure is explained considering all process parameters and design calculations, methodology used, material used for moulding, mould steels recommended and their properties.

Keywords: hand guard, Pro-E and Plastics Advisers

Introduction
Now a day's plastics are occupying a vital role in the day-to-day life. It is not at all can exaggeration, if we say that there is no field into which plastics have not stepped in right from the manufacturing at a pin to that of a spacecraft [1]. At the middle of 19th century, plastics start a leading role in the material and in our life. Easy availability of raw material, versatility in manufacturing esthetic appearance, lightness in weight, corrosion resistance are some of the aspects through which plastics are even becoming superior to metals and attained an elevated rate of preference in every branch of manufacturing. Right from the emergency of plastics they are undergoing drastic in various aspects like designing of plastics products, manufacturing processing and then in testing fields and now, because of fruitful efforts of many people at last the day came in which we completely manufacturing a plastic products without utilizing much work through manual labor, but with the help of artificial intelligence that is, with software package like CAD &CAM. Because of high strength to weight ratios, improved chemical and temperature resistance, inherent properties of being thermal and corrosion resistance, transparency have made them a material choice [2]. Plastics consume less energy during formation and can be profitably recycled. Today plastics are replacing the metals like Brass, Copper, Cast-iron, Steels etc.

Objectives
- To study the plastic injection moulding process
- To produce the quality of plastic hand guard product with no defect
- To get the optimum value for parameter setting of plastic injection molding
- Process (hand guard mould)

Scope of Study
- Study the procedure/method to set up the mould in injection moulding machine
- Study the parameters of plastic injection moulding process
- Material Selection
- Study the types of defects occurs to the plastic hand guard product
- The method for DOE
- The procedure for experiment
- Analysis Result.

Outline of Study
This research will be divided into six chapters. The first chapter is mainly about the introduction of the research, problem statement. Objectives, scope of study, importance of study and lastly about the study outlines. The next chapter is the literature review. Based on the reference gathered (journals, books, websites etc.), this chapter will discuss the definition and introduction to plastic injection moulding. It will also discuss the method on how the research will be done based on the past researcher.

General Properties of Plastics
- Light weight
- Thermal insulation
- Electrical insulation
- Toughness
- Resistance to fatigue
- Corrosion possibilities
- Coloring possibilities
- Water resistance
- Greater design freedom

- Chemical resistance
- High strength at low temperature
- Low cost of production
- Better strength
- Low cost
- Easy machine ability
- Resistance to corrosion environment wears chemicals
- Thermal insulation
- Toughness

Plastics can be classified according to manufacturing methods in to two main groups:

1. Those which soften when heated and solidifies on cooling. These are known as “thermoplastics” and they are recyclable because they only undergo temporary physical change when heated. When the heat is removed they return to their original state. This is known as heat softening [3].
2. Those which harden when heated as a result of chemical change. These are known as “thermo sets” and they are not recyclable because they undergo a permanent chemical change when heated. This is known as heat hardening. Relative Advantages of Product, when made with plastic material easy to mould products with considerable shapes can be easily produced, provided that the configuration allows the products to be ejected from the mould. Furthermore, the injection moulding allows for moulding a finished product through a single process. Thus, plastic moulding is for more efficient than metal work [4]. Freely colored and made opaque or transparent material resins can be freely colored by adding colorants and can give a glossy transparent finish as desired, these by improving the product quality. Light yet strong products are obtainable, while plastics are lighter than metals or porcelain, some plastic products have better mechanical properties. Especially plastic product containing glass fiber features a high physical strength. Besides, the lightness considerably reduce the overall weight of the product using plastic components [5].

Free from rust and corrosion

Generally speaking plastics can be endure various chemicals so they do not rust or

Corrode as metals do and are not affected by oils, chemical.

- It is possible to make it as a single component only with plastic material.
- We can get the component at relatively low cost, once the moulding comes out of the mould, we can use it directly without any finishing operations [6].

Component Details

The component hand guard is used to help handicapped peoples, which is helpful to walk easily. The component is used to medical field. The material of the component is polypropylene and the component much is free from flow marks, flashes and weld lines.

Component Name	: Hand guard
Component Weight	: 66 gm
Density	: 0.90 gm/cm^3
Max. Wall thickness	: 5mm
Material	: polypropylene
Shrinkage	: 0.010-0.025%

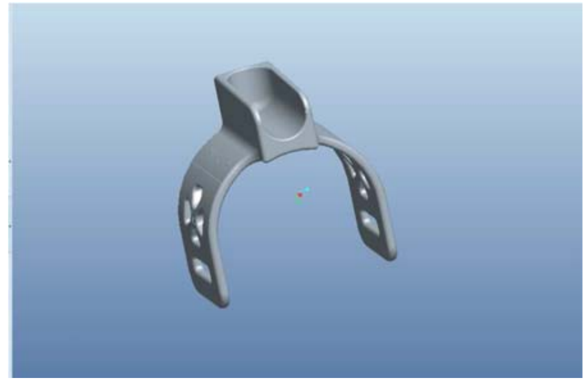

Fig 1: Sample component

Mould Design Hand Guard

Name of the component: Hand Guard

Material	: polypropylene
Shrinkage	: 0.010-0.025%
Density	: 0.90g/cm^3
Wt of the component	: 66gms
Distance b/w tie bars	: 460x460 mm
Injection pressure	: 2200 kg/cm^2
Thermal conductivity of plastic	: 3.3x10-4 cal/sec.cm
Mould weight max	: 1100 kgf

Moulding Machine Specifications & Design Calculations

1) Projected area of the component

PA = (PA1+PA2) x No. of impressions

= (l x b) + (l x b) x1

= (109.70 x 50.13) + (31 x 40.66) x1

=6759.72 mm^2

=6759.72/100

=67.59 cm^2

2) Clamping tonnage = TPA x (2/3 of average injection pressure)

=67.59 x (2/3)2200

=67.59 x 1466.66

=99132 kg/cm^2

=99132/1000

=99.132 tons

3) Component weight = (component volume) x (material density)

=73444.6 x 0.90

=66100.14

=66100.14/1000

=66 gms

Moulding machine specifications

Clamping unit	: 125 tons
Overall size of plantes	: 700x700 mm
Distance b/w the tie bars	: 460x460
Mould height min	: 220 mm
Mould height max	: 550 mm
Mould dia min	: 260 mm
Mould weight max	: 1100 kgf
Mould wt on moving platen max	: 700 kgf

4) Shot weight of component= weight of component x no. of impressions

=66 x 1

=66 gm

Methodology of Mould Design

- Check the model feasibility to design
- Identify critical and major dimensions
- Decide parting line
- Add drafts to the model considering tolerances for maximum material condition
- Add shrinkage to the model
- Generate surface as per the decided parting line
- Split core and cavity
- Internal splitting of core and cavity inserts considering manufacturing and assembly
- feed back
- Check for draft analysis and clearance analysis
- Feed system creation as per the standard or customer requirement
- Create mold base
- Provide ejector positions in the component and inserts
- Create cooling holes
- Conduct concept design review meeting
- Change the model or layout based on the review output
- Conduct assembly review meeting
- Prepare detail drawings of inserts and mold base elements
- Release drawings to manufacturing
- Collect the document distribution list
- If any changes in the drawings then new drawings must need to be released with new
- revision
- Bill of materials and revision record to updated

Mould Flow Analysis

Now a day's the technology of the tool and die fabrication in plastic injection is one of the world's fastest growing industries. Plastic is now used in almost every application, ranging from household articles to space travel, from transportation to packing, from medicine to toys, from bridge building to sports. Generally, injection molding is a process that terms the plastic into a desired shape by melting the plastic material under pressure into the mould cavity. The shape of the plastic that is desired is achieved.

Result and Discussions

Product Simulation

Before starting the simulation process, some information has to be set up, such as material supplier and material grade. This is normally already set in the Mold flow database and the user just selects the specific material specification.

Solid Model

Solid model shows the sprue location placing on the top of the model. The yellow colour cone shape shows the injection location in this analysis.

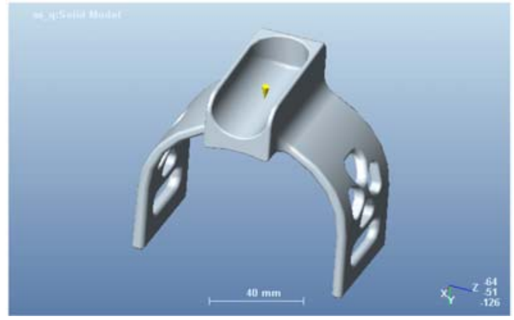

Fig 2: solid model

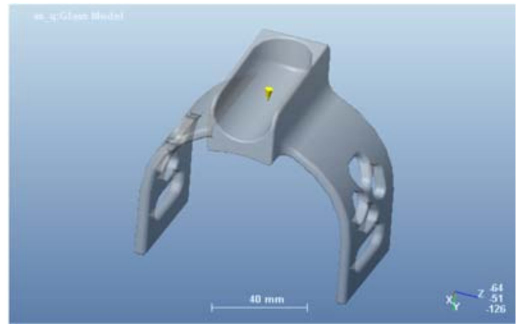

Fig 3: Glass model Glass model shows clear view of particular component.

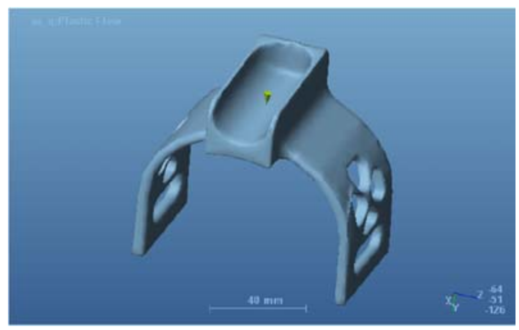

Fig 4: plastic flow

The fill pattern will show the designer if and how the part fills, and helps the designer understand how weld lines and air traps will form. Other problems, such as over packing and hesitation, can be readily identified by the animated analysis.

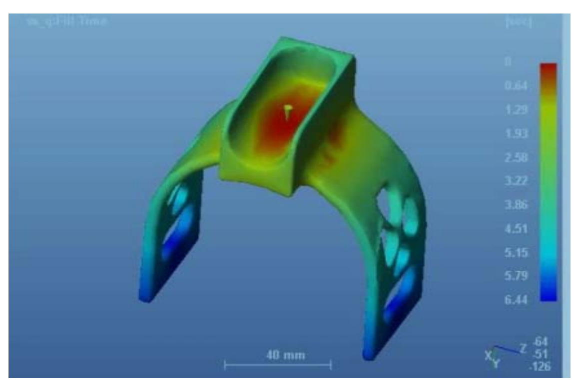

Fig 5: Fill time

Fill time is the time taken to fill up the part inside the cavity; it is also to show how the Plastic material flows to fill the mould. From that we know that the short shot (part of the model which did not fill) part will be displayed. From that result one can also understand how the weld line and air trap will form. Above figure shows the material filling into the mould.

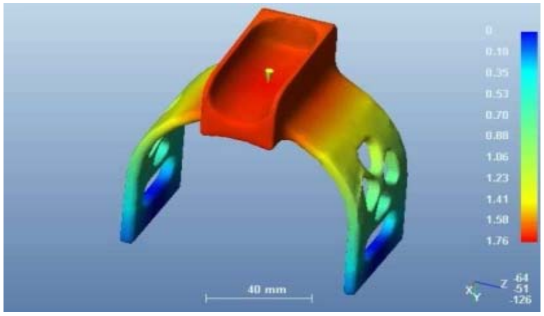

Fig 6: Injection pressure

The color at each place on the model represents the pressure at the place on the model. Two colors show the highest pressure (red) and lowest pressure (blue). The Injection pressure can be used in conjunction with pressure drop result. For example, even if a section of a part has an acceptable pressure drop, the actual injection pressure in the same area may be too high. High injection can cause over packing. To Reduce the chance of this happening, follow these steps:

1. Increase the maximum injection pressure
2. Alter the polymer injection location
3. Alter part geometry
4. Select a different material

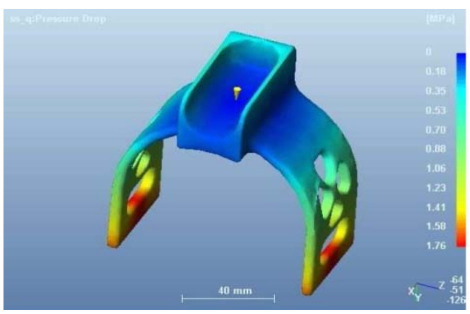

Fig 7: pressure drop

Above figure shows the pressure drop result. When the place is filled, the pressure will drop from the injection location to the end of the filled part. The pressure drops result uses a range of colors to indicate the region of highest-pressure drop. The pressure drop is one Factor used to determine the confidence of fill result.

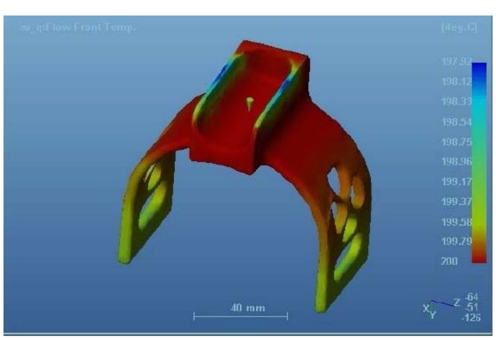

Fig 8: Flow front temperature

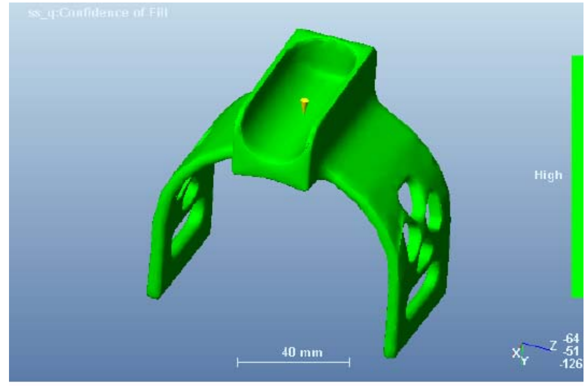

Fig 9: confidence of fill

The confidence of fill result displays the probability of the region within the cavity with plastic. This result is derived from the pressure and temperature results. The confidence of fill will display in three colors: green, yellow, red and translucent. Showing in green will definitely fill, in yellow may be difficult to fill or may have quality problem, in red will be difficult to fill or will have quality problem and in translucent will not fill. If the cavity does not fill, some changes must be made to either design, gate location, Choice of plastic or processing conditions. However, to ensure the finished part is of Good quality, the cavity must also be adequately packed with plastic as shows Confidence of fill in this analysis.

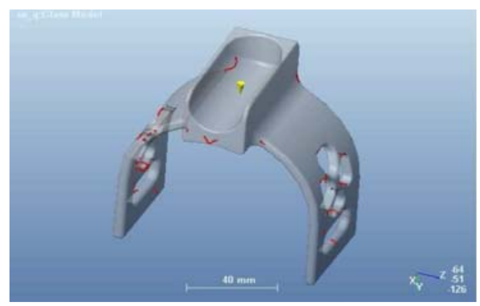

Fig 10: weld lines

Weld lines are created when two or more cooling melt flow fronts meet within the mould. This can be recognized on a moulding as a hairline feature and occurs where melt flow has been divided around an obstacle in the tool, such as a boss pin, and rejoins on the other side. Weld lines locally reduce the mechanical properties of the material at that point and care should be taken to position gating such that weld lines are minimized. If they are unavoidable, they must be positioned in areas of least effect. Melt flow software packages are of great assistance in this area for complex mouldings. Modification of process conditions such as increasing melt temperature, mould temperature or injection speed may improve the situation but may create other problems. Problem suggested corrective action Weld lines:

1. Increase injection pressure.
2. Increase packing time/pressure.
3. Raise mould temperature.
4. Raise material temperature.
5. Vent the cavity in the weld area.
6. Provide an overflow well adjacent to the weld area.
7. Change gate location to alter flow pattern.

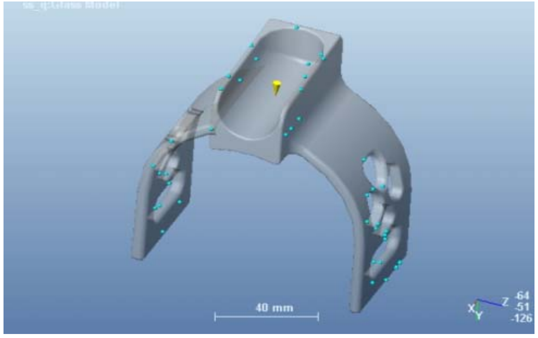

Fig 11: air traps

The small blue bubbles are showing the air traps in the parts. Air traps result shows the regions where the melt stops at a convergence of at least 2 flow fronts or at the last point of fill, where a bubble of air becomes trapped. To prevent air traps occurring when converging flow fronts surround and trap a bubble of air, balance flow paths by either:

1. Using flow leaders / deflectors
2. Changing part wall thickness
3. Changing polymer injection locations

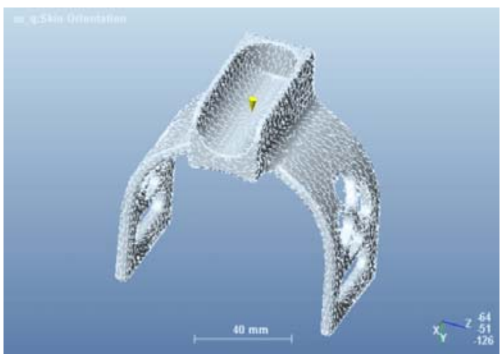

Fig 12 skin orientation

Problem Suggested Corrective Action(s)

1. Increase injection pressure.
2. Increase packing time/pressure.
3. Use maximum ram speed.
4. Raise mould temperature (voids).
5. Lower mould temperature (sinks).
6. Decrease cushion.
7. Increase size of sprue and/or runners and/or gates.
8. Relocate gates nearer thick sections.

Conclusions

The dies have been designed by using pro-E and auto cad software's which are very handy to expedite the design faster various factors were considered to produce a defect free and economical plastic component during the design stage itself. The injection of the component is much easier the cost and cycle time is reduced. This design proves to be economical. From the analysis simulation, Mould flow provides sufficient information results such as fill time, injection pressure and pressure drop. With this result, users can avoid the defect of the plastic in actual injection such as sink mark, hesitation, air traps, and over packing. The analysis will also help the mould designer to design a perfect mould with minimum modifications and it will also reduce the mould setup time. With this analysis and simulation it will help to reduce time and cost. From the above results one can conclude that mold design and used material are safe and suitable for the respective moulding process.

Future Scope

In my project mould flow analysis will used. By using this analysis we can find confidence to fill, filling time, injection pressure. In this one spure location will selected. In future by using two sprue locations we can increase the production rate. If anybody wants to simulate molten flow and wants to know stresses and displacements, one has to perform transient thermal analysis supported by computational fluid dynamic analysis.

References

1. ASM Handbook Volume 15, mould (Edition), 2008.
2. Injection moulds design RGW PYE, Joseph dym (Edition), 2006.
3. Plastic Materials I.A Bryson (Edition), 2006.
4. Applications of plastics JS Anand (Edition), 2007.
5. Theory of Moulds AMSCO Publications (Edition), 2002.
6. Mould designs Manual IDTR Jamshedpur (Edition), 2008.